Paper submitted to Thermal Fluid Analysis Workshop (TFAWS) to be hosted by NASA Glenn Research Center and Corporate College, Cleveland, September 10-14, 2007

Modeling of Compressible Flow with Friction and Heat Transfer using the Generalized Fluid System Simulation Program (GFSSP)

Alak Bandyopadhyay & Alok Majumdar
Alabama A&M University
Huntsville, Alabama 35762
NASA/Marshall Space Flight Center
Huntsville, Alabama 35812

Abstract

The present paper describes the verification and validation of a quasi one-dimensional pressure based finite volume algorithm, implemented in Generalized Fluid System Simulation Program (GFSSP), for predicting compressible flow with friction, heat transfer and area change. The numerical predictions were compared with two classical solutions of compressible flow, i.e. Fanno and Rayleigh flow. Fanno flow provides an analytical solution of compressible flow in a long slender pipe where incoming subsonic flow can be choked due to friction. On the other hand, Raleigh flow provides analytical solution of frictionless compressible flow with heat transfer where incoming subsonic flow can be choked at the outlet boundary with heat addition to the control volume. Non uniform grid distribution improves the accuracy of numerical prediction. A benchmark numerical solution of compressible flow in a converging-diverging nozzle with friction and heat transfer has been developed to verify GFSSP's numerical predictions. The numerical predictions compare favorably in all cases.

Introduction

Most commercial network flow analysis codes lack the capability to simulate one-dimensional flow in a rocket engine nozzle. Simulation of one-dimensional flow in rocket nozzle requires a numerical algorithm capable of modeling compressible flow with friction, heat transfer, variable cross-sectional area and chemical reaction. One of the primary requirements of compressible flow simulation is to accurately model the inertia force which is often neglected in many network flow analysis codes. NASA Marshall Space Flight Center has developed a general purpose finite volume based network flow analysis code: Generalized Fluid System Simulation Program (GFSSP) [1] which is widely used for the design of Main Propulsion System of Launch Vehicle and secondary flow analysis of turbopump. The purpose of the present paper is to verify and validate GFSSP's numerical predictions with several benchmark solutions described in the following section.

Problem Description:

In this study, mainly two types of geometries are considered – (a) a straight pipe of constant diameter, and (b) a converging-diverging nozzle of linearly varying diameter.

The effect of friction and heat transfer on the pressure, temperature and Mach no. are studied using these two geometries. The problems are divided into five different cases, namely:

Case No.	Description
1.	Fanno Flow – flow with friction in an adiabatic constant area pipe.
2.	Rayleigh Flow – flow with heat transfer in a frictionless constant area pipe.
3.	Combined Friction and Heat Transfer in a constant area pipe.
4.	Effect of friction and area change using an adiabatic converging-diverging nozzle.
5.	Combined Friction and Heat Transfer in the converging-diverging nozzle.

For each of these problems the flow is assumed to be one dimensional and pressure, temperature and Mach number is evaluated using analytical and numerical (Generalized Fluid System Simulation Program) methods.

Constant Area Duct

For the first three problems, geometry is the same, a constant area pipe, as shown in the Figure 1 as given below.

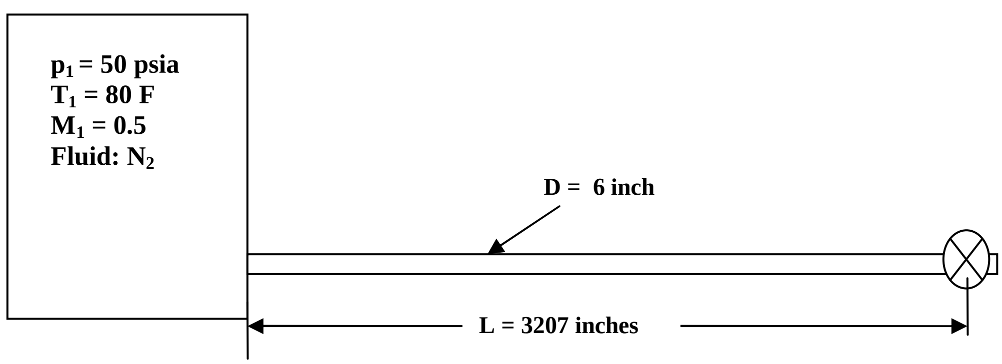

Figure 1 Schematic diagram for a constant area pipe.

The pipe is assumed to have a constant friction factor (f). The length of the pipe is chosen so that the flow becomes choked at the exit of the pipe as determined from analytical solution.

Converging-Diverging Nozzle

A schematic diagram for a converging-diverging nozzle is shown in figure 2. It is assumed that the diameter of the nozzle is changing linearly, and it is at the lowest at the throat. The dimensions are arbitrarily chosen and might not resemble a realistic nozzle. The objective of the present work is to validate GFSSP with analytical solution. The nozzle is assumed to have a constant friction factor (f) and constant heat flux (q). Three different subproblems will be considered – flow with friction only (case 4), flow with heat transfer only (case 5) and both friction and heat transfer (case 6).

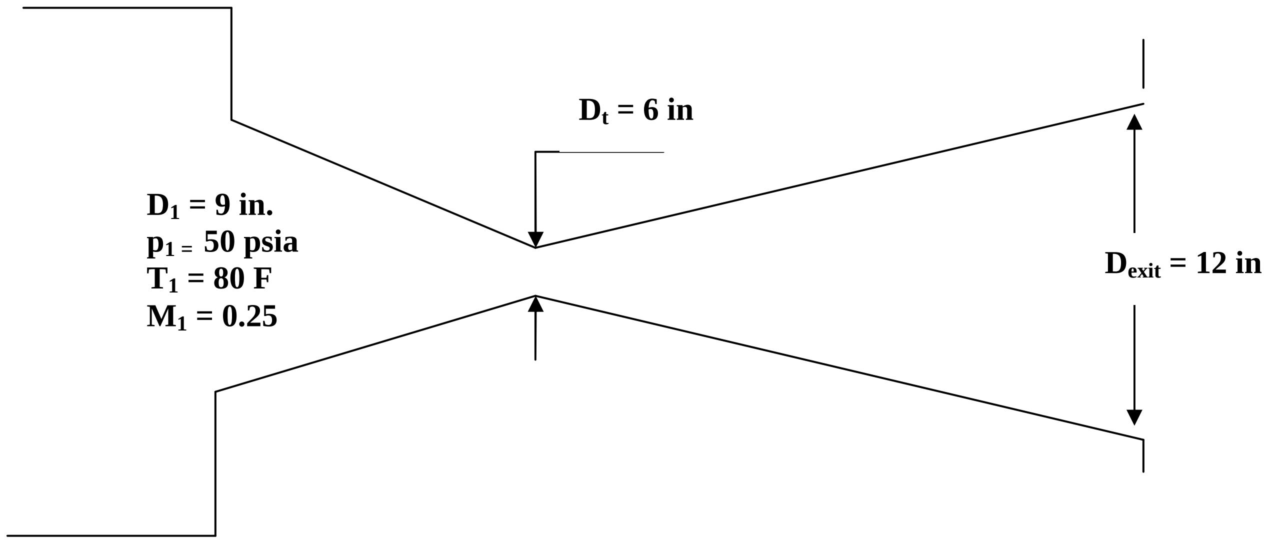

Figure 2 Schematic of a converging-diverging nozzle

Benchmark Solutions

The generalized one dimensional compressible flow can be described mathematically using the following conservation equations. These equations are applicable to study the combined effect of area change, friction and heat transfer in a converging-diverging nozzle as well as in a constant area duct.

The equations are in the differential form.

Mass Conservation:

$$\frac{d\rho}{\rho}+\frac{dA}{A}+\frac{dV}{V}=0 \tag{1}$$

Momentum Conservation:

$$\frac{dp}{p}+\frac{\gamma M^2}{2}\frac{fdx}{D}+\gamma M^2\frac{dV}{V}=0 \tag{2}$$

Where, M = Mach no. = $\frac{V}{C} = \frac{V}{\sqrt{\gamma \frac{p}{\rho}}}$

Using the definition of Mach number, Stagnation temperature, Equations (1) and (2) can be expressed as an ordinary differential equation of 1st order.

$$\frac{dM}{dx} = \frac{M\left(1+\frac{\gamma-1}{2}M^2\right)}{\left(1-M^2\right)}\left[\gamma M^2 \frac{f}{D} + \frac{\left(1+\gamma M^2\right)}{2T_0}\frac{dT_0}{dx} - \gamma M^2 \frac{1}{A}\frac{dA}{dx}\right] \qquad (3)$$

With boundary value, $M(x = 0) = M_1$ (3b)

$\frac{dT_0}{dx}$ in equation (3) can be determined from energy equation which can be expressed as:

$$q(\pi D)dx = \dot{m}\, c_p dT_0 \qquad (4)$$

Given the inlet conditions 3(b), the 1st order differential equation (eqn. 3) in M is solved to find the Mach number at any x location. As this equation is a nonlinear equation in M this equation is solved by using 4th order Runge-Kutta method [2].

From Mach number, the static temperature and pressure can be determined from the following equations:

$$\frac{T(x)}{T(0)} = \frac{T_0(x)}{T_0(0)} \frac{1+\frac{\gamma-1}{2}(M(0))^2}{1+\frac{\gamma-1}{2}(M(x))^2} \qquad (5)$$

$$\frac{p(x)}{p(0)} = \frac{A(0)}{A(x)}\frac{M(0)}{M(x)}\sqrt{\frac{T(x)}{T(0)}} \qquad (6)$$

Numerical Modeling

GFSSP employs a finite volume formulation of mass, momentum and energy conservation equations in a network consisting of nodes and branches [3]. Energy conservation equations are expressed in terms of entropy with entropy generation due to viscous dissipation. Mass and energy conservation equations are solved for pressures and entropies at the nodes while the momentum conservation equation is solved at the branches to calculate flow rate. The friction in pipe was modeled by Darcy friction factor. The pressure drop in a pipe is expressed as:

$$\Delta p = k_f \dot{m}^2 \tag{7}$$

$$\text{where, } K_f = \frac{8fL}{\rho_u \pi^2 D^5 g_c} \tag{7b}$$

For the numerical simulation using GFSSP, the entire domain (pipe or the nozzle) is divided into several sectors, and each of these sectors is represented by a pipe of constant diameter. The diameters for adjacent sectors will vary for converging-diverging nozzle. The fluid is assumed to be Nitrogen. The numerical solutions generated by GFSSP is presented in the results and discussion section and compared with the analytical solutions.

Discretization

For the numerical simulation, the pipe in Figure 1 is divided into finite number of pipes of non-uniform length as shown in Figure 3. A node is being placed at the end of each pipe sectors including the two boundary nodes. Both uniform and non-uniform node distributions have been tried for the simulations. It has been observed that a total of 21 nodes with clustered nodes near the inlet and exit (figure 3), is sufficient for the constant area pipe (cases 1 through 3) to get grid independent solution.

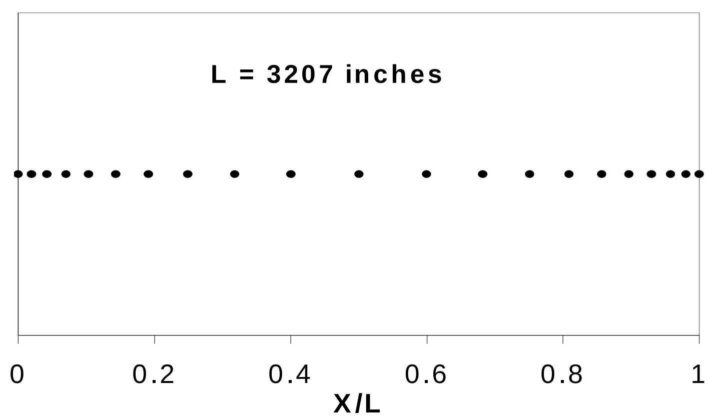

Figure 3. Non-uniform node distribution for the constant area duct.

Results & Discussion

The analytical solution for all the cases are obtained from the generalized numerical solution of equation (3) and these solutions have also been reproduced and verified with Fanno and Rayleigh tables available in any standard text book [4]. Cases 1 through 3 are for constant area ducts and cases 4 and 5 are for variable area ducts (nozzle flow).

Case 1: Fanno Flow:

Flow with friction, but no area change and heat transfer. No heat transfer implies the stagnation temperature is constant and $dT_0/dx = 0$.
Before presenting the results for Fanno flow, choice of pipe length as 3207 inches is explained as below.

From the analytical solution for Fanno flow [4], the critical length of the pipe (L^*) is determined from the following equation:

$$\frac{fL^*}{D} = \frac{1-M^2}{\gamma M^2} + \frac{1+\gamma}{2\gamma}\ln\left(\frac{(1+\gamma)M^2}{2+(\gamma-1)M^2}\right) \tag{7}$$

M is the inlet Mach number. The critical length of the pipe is the length required for the flow to choke at the exit (i.e. M = 1 at exit). With an inlet Mach no of 0.5 and a friction factor of 0.002, and pipe diameter of 6 inches the critical length is calculated to be 3207 inches. This length is kept fixed for the cases of constant area pipes.

Figure 4 shows a plot of the p/p^* ratio with different types of node distribution for the numerical solution compared with the analytical solution. A non-uniform node distribution with a total of 21 nodes (20 control volumes) is sufficient to get a grid-independent solution. The plot also shows that the numerical solution using GFSSP agrees very well with that of the analytical solution.

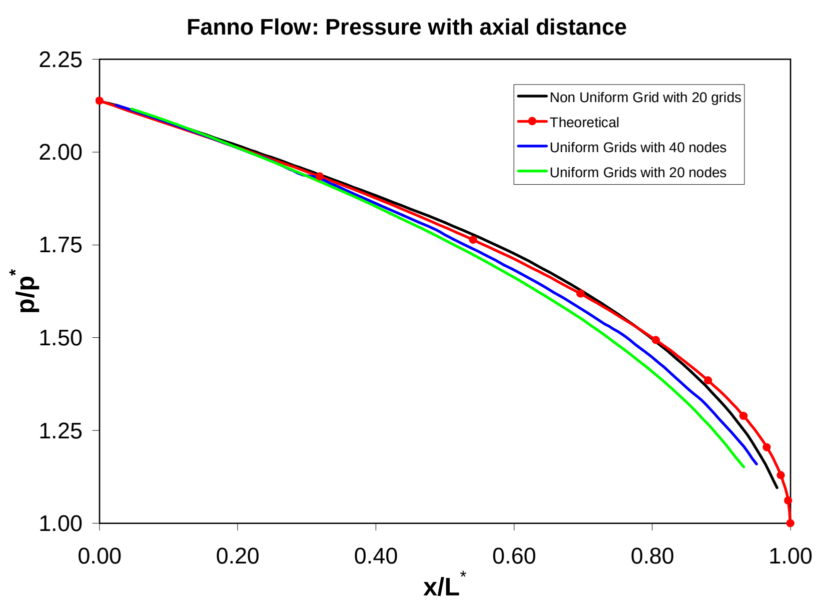

Figure 4. Pressure distribution for Fanno Flow with various grid distributions.

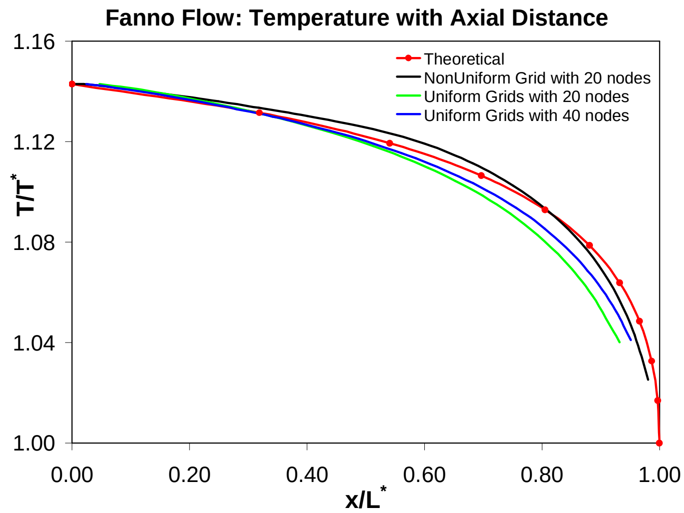

Figure 5. Temperature distribution for Fanno Flow with various grid distributions.

The corresponding temperature distribution is shown in figure 5.

The Mach no. is a derived quantity for the numerical solution and it is calculated from the mass flow rate, temperature and pressure as follows.

$$M = \frac{V}{C} = \frac{4}{\pi}\sqrt{\frac{R}{\gamma g_c}}\,\frac{\dot{m}\sqrt{T}}{pD^2} \tag{8}$$

Where, R = gas constant = 55.19 ft-$lb_f/(lb_m R)$, g_c = 32.17 ft/sec^2. p is in psia, T in Rankine and D is the diameter in inches, $\dot{m}$ is the flow rate in lb_m/s. Figure 6 show a plot of the Mach no. along the axial direction, and again the agreement with the analytical solution is quite good. The slight difference even at the inlet is because, in GFSSP the mass flow rate is not prescribed, rather the pressure boundary condition is specified, and the flow rate is computed.

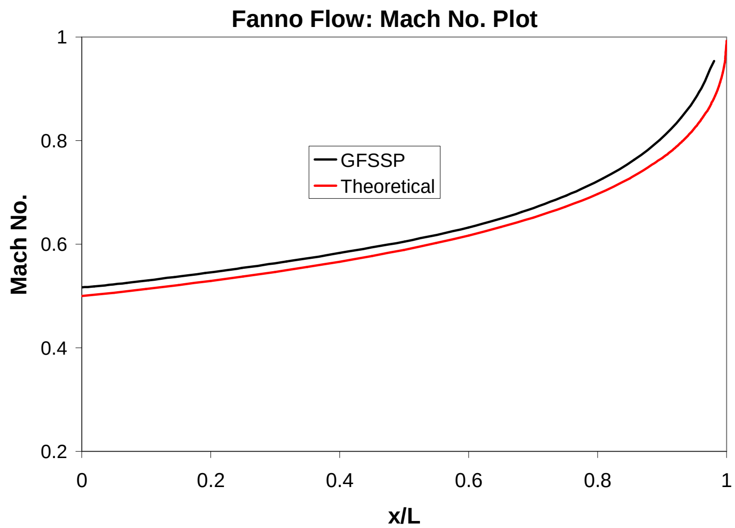

Figure 6. Case1 – Fanno Flow. Plot of Mach number along the pipe length.

Case 2 – Rayleigh Flow: Flow with no friction and a uniform heat transfer

Figures 7, 8 and 9 show the distribution of temperature, pressure and Mach number for a heat input rate of 2088 Btu/sec on the same pipe geometry that has been used for Fanno flow. The inlet Mach number is chosen as 0.46 and it has been analytically calculated that with this inlet Mach number, a heat input of 2088 Btu/sec makes the flow choked at the exit (i.e. Mach number = 1). Further increase in the heat rate will make the flow supersonic and in the present work, only subsonic flow is being considered. Again, the numerical results show quite good agreement (within 5%) with the analytical solution.

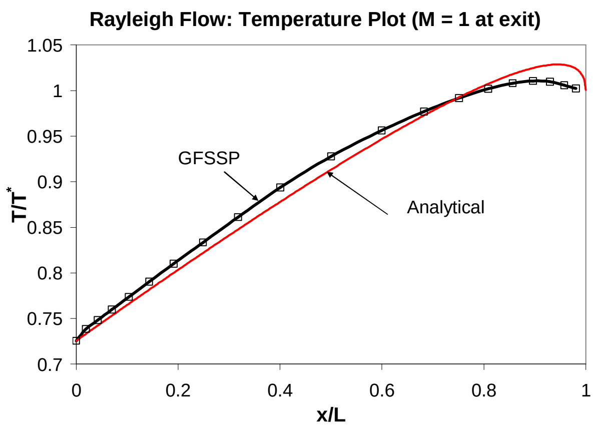

Figure 7: Temperature distribution for Rayleigh Flow (Q = 2088 Btu/s)

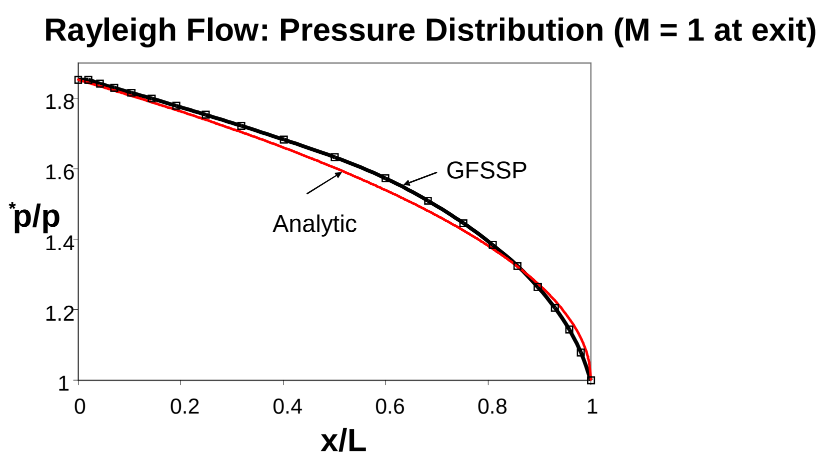

Figure 8. Pressure distribution for Rayleigh Flow (Q = 2088 Btu/s).

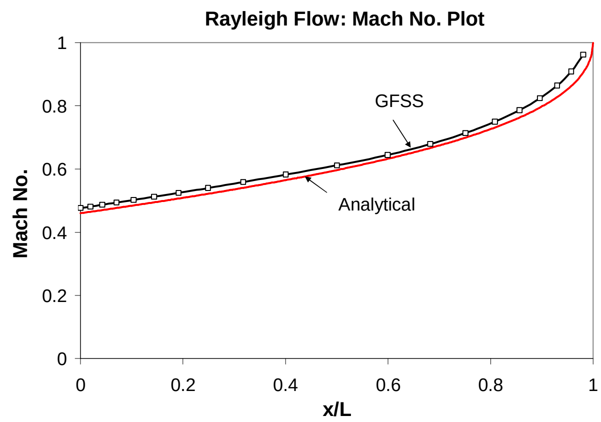

Figure 9. Mach no. distribution for Rayleigh Flow (Q = 2088 Btu/s).

Case 3: Combined Friction and Heat Transfer in a constant area Pipe

There is no standard table available for the combined effect of friction and heat transfer, and analytical solution can only be obtained by solving the differential equation in Mach number (eqn. 3). Figure 10, 11 and 12 show the combined effect of friction and heat transfer on the temperature, pressure and Mach no. respectively. The inlet Mach no. is 0.45. All three plots correspond to f = 0.002, Q = 555 Btu/sec. The pressure and temperature are plotted as dimensional quantities, with inlet pressure as 50 psia and inlet temperature of 80 F.

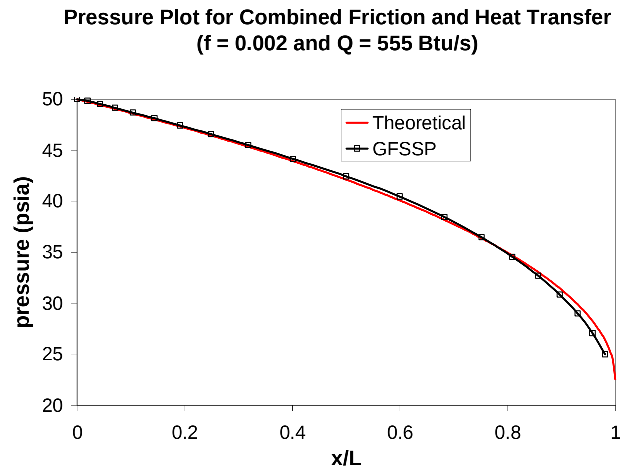

Figure 10. Effect of friction and heat transfer on the Temperature.

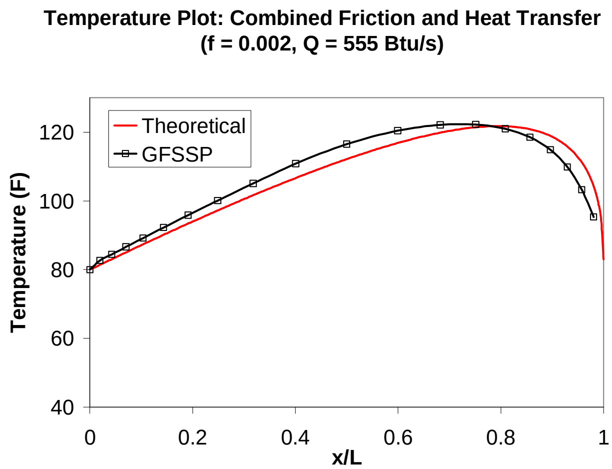

Figure 11. Effect of friction and heat transfer on the Pressure.

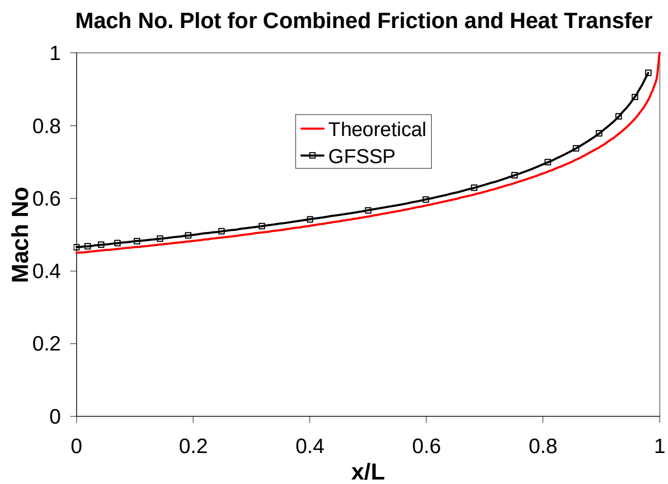

Figure 12. Effect of friction and heat transfer on the Mach number.

Case 4 and Case 5: Nozzle Flow with Friction Only, and both Friction with Heat

The nozzle dimensions and operating conditions are given in figure 2. The wall friction factor is taken as 0.05. The nozzle is assumed to be adiabatic. For the numerical simulation the pressure and temperature at the inlet are specified as given and exit pressure is specified. The exit pressure is calculated from the analytical solution to ensure that the inlet Mach number is 0.25.

Figure 13 shows the effect of node distribution for the numerical simulation and it is observed that about 64 nodes, mostly uniform, except clustered near the inlet and throat, produces grid independent solution. The numerical solution agrees well with the analytical except near the throat.

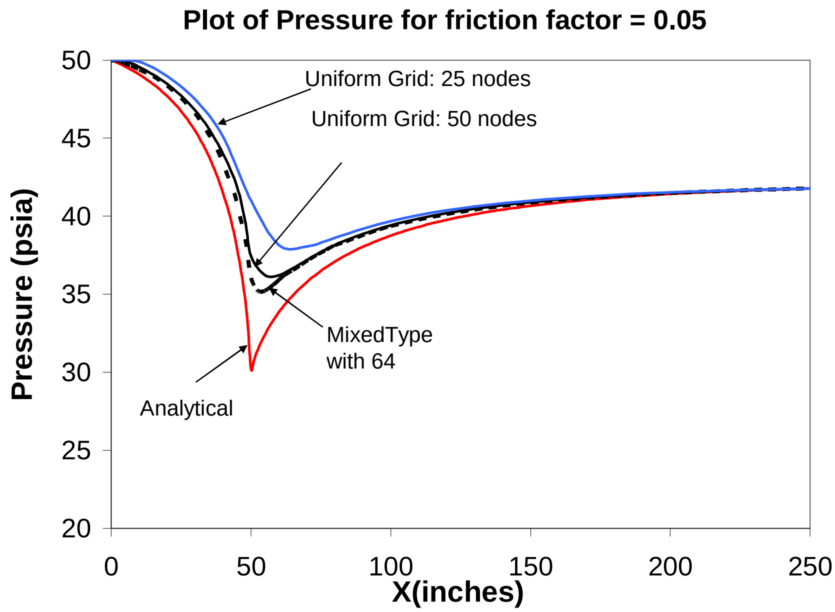

Figure 13. Grid study on the pressure distribution in a converging-diverging nozzle.

Figure 14 and 15 show the effect of friction factor and both friction factor and heat transfer on the Mach number distribution and the pressure distribution respectively.

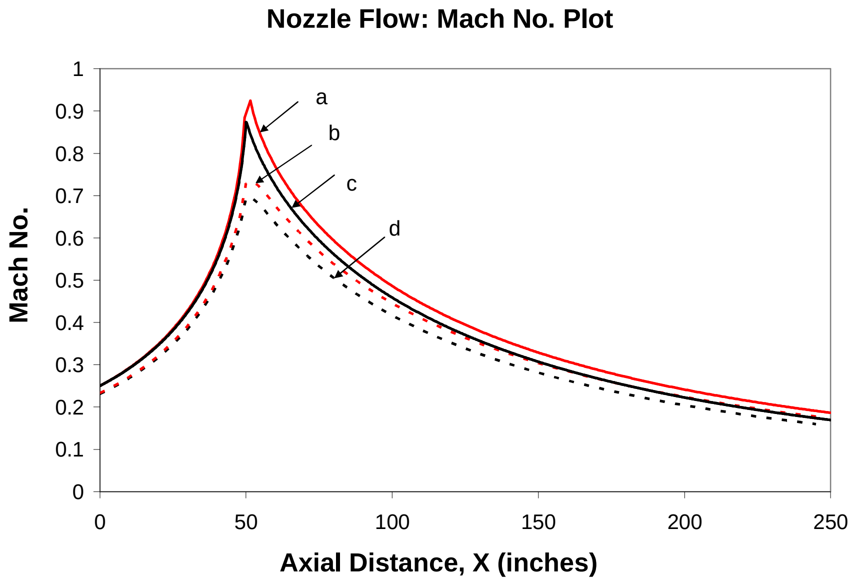

Figure 14. Plot of Mach number for nozzle flow – (a) Effect of friction and heat transfer – analytical solution (b) Effect of friction and heat transfer – numerical (GFSSP) solution (c) Effect of friction only – Analytical and (d) Effect of friction only – numerical simulation.

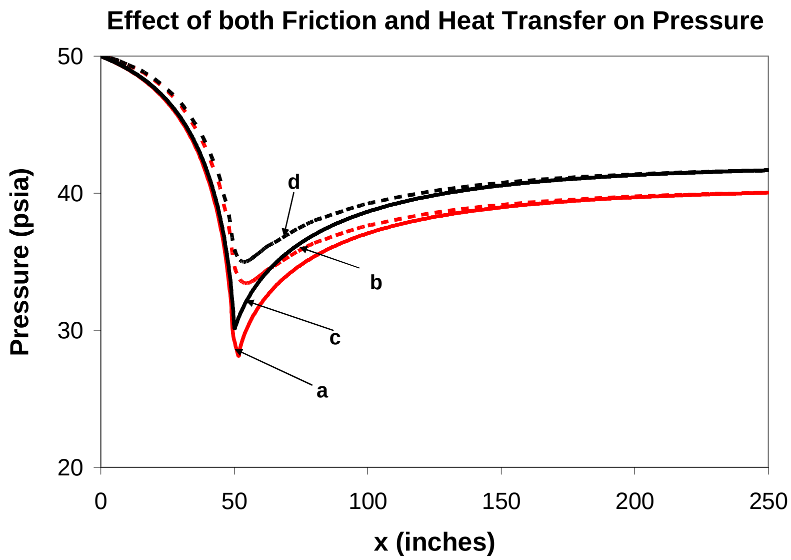

Figure 15. Pressure distribution for nozzle flow with (a) Both friction and heat transfer – Analytical (b) Both friction and heat transfer – Numerical (GFSSP) (c) Friction only – Analytical and (d) Friction only – Numerical (GFSSP).

Both of these plots show very good agreement for most of the locations between the numerical and analytical. Figure 16 shows the corresponding temperature plot.

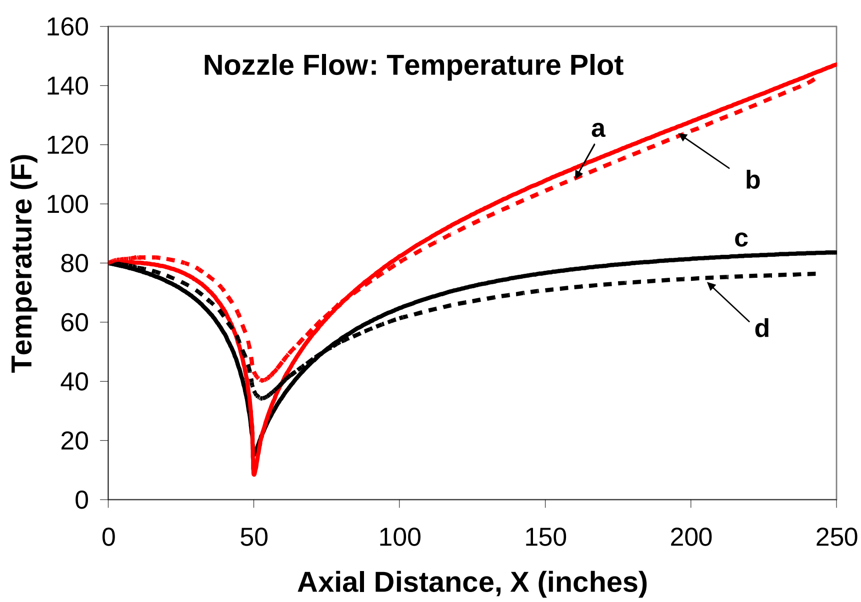

Figure 16. Temperature distribution for nozzle flow with (a) Both friction and heat transfer – Analytical (b) Both friction and heat transfer – Numerical (GFSSP) (c) Friction only – Analytical and (d) Friction only – Numerical (GFSSP).

All of these plots show a very good agreement between analytical and numerical results.

Conclusions

The paper presents a numerical study of the effect of friction, heat transfer and area change in subsonic compressible flow to verify the accuracy of pressure based finite volume algorithm implemented in Generalized Fluid System Simulation Program. The numerical solutions of pressure, temperature and Mach number have been compared with benchmark solution for five different cases representing the effect of friction, heat transfer and area change. Generally there is a good agreement between the numerical solution and benchmark solution. It has been observed that non-uniform grid improves the accuracy of numerical solution. The observed discrepancy at the throat of the converging-diverging nozzle is due to sharp discontinuity at the nozzle throat which has not been accounted for in the numerical model where the nozzle has been discretized by a series of pipes with varying cross-sectional area. The modeling of supersonic flow is being planned for future investigations.

Acknowledgement

This work has been performed under NASA/Marshall Space Flight Center Summer Faculty Research Program (2007) in the Thermal and Combustion Analysis Branch (ER43) of Propulsion System Department in Engineering Directorate. The authors would like to thank Mr. Bruce Tiller for his support and encouragement.

References

1. Majumdar, Alok, Steadman, Todd and Moore, Ric, "Generalized Fluid System Simulation Program (GFSSP) Version 5.0." George C. Marshall Space Flight Center, Engineering Directorate Technical Report (To be published as NASA Technical Memorandum), February, 2007

2. Press W.H. et al., “Numerical Recipes in Fortran: The Art of Scientific Computing”, 2nd Ed.,1992, Cambridge University Press, pp 704-716.

3. Majumdar, A.K., “A Second Law Based Unstructured Finite Volume Procedure for Generalized Flow Simulation”, 37th AIAA Aerospace Meeting Conference and Exhibit, January 11-14, 1999, Reno, NV, AIAA 99-0934.

4. Saad, Michael A, “Compressible Fluid Flow”, 2nd Edn., 1993, Prentice Hall Publications.

Nomenclature

Alphabets:

A	Area of the pipe, or local area of the nozzle at any axial location.
C	speed of sound

c_P Specific heat at constant pressure.
D local diameter
f Darcy Friction Factor
L Length of the Pipe, Nozzle
M Mach number
$\dot{m}$ mass flow rate
p Pressure
Q Total Heat Transfer Rate
q Heat Flux
R Gas Constant
T Temperature
V Velocity

Greek Symbols:

ρ Density
γ Specific Heat Ratio

Subscripts:

1 Inlet
0 Stagnation Properties
t at the throat of the nozzle.

Superscripts:
* Choked Properties (Corresponding to M = 1)